International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)
ISSN(P): 2249-6890; ISSN(E): 2249-8001
Vol. 7, Issue 1, Feb 2017, 53-60

COMPUTATIONAL ANALYSIS OF A CD NOZZLE WITH 'SED' FOR A ROCKET AIR EJECTOR IN SPACE APPLICATIONS

CH V K N S N MOORTHY[1], V SRINIVAS[2], V V S H PRASAD[3] & T VANAJA[4]

[1,3,4]Department of Mechanical Engineering, Institute of Aeronautical Engineering (Autonomous) Dundigal, Hyderabad, Telangana, India

[2]Department of Mechanical Engineering, GITAM University, Rushikonda, Vishakhapatnam, Andhra Pradesh, India

Original Article

ABSTRACT

The main objective of the work presented in this paper is to optimize the configuration of rocket air ejector. An ejector system is to be designed in order to pump out the gasses from low pressure to ambient conditions. Ejectors mainly use the principles of fluid dynamics for pumping. They do not consist of any mechanical parts and hence no wear and tear. But consequently the design of the ejectors should be very much precise for the proper and reliable function. In this part of the work, the configuration of a CD nozzle with 'SED' is calculated from a predefined rocket air ejector configuration. 3D models are developed using AutoCAD, meshed and analyzed using Ansys Cfx. Taking the predefined input and boundary conditions, pressure, temperature, Mach number and velocity contours are developed for the analysis to identify the convergence. Parametric analysis is also carried out by plotting various graphs to understand the corresponding effect.

KEYWORDS: CFD, Ejector, Nozzle, SED & Flow Analysis

Received: Dec 24, 2016; **Accepted:** Jan 21, 2017; **Published:** Jan 31, 2017; **Paper Id.:** IJMPERDFEB20176

INTRODUCTION

Upper stage rocket motor testing is the most vital, difficult and involves a lot of complexity. The decrease in the atmospheric pressure at higher altitudes affects the performance of launch vehicles. Selection of perfectly suitable fluid dynamic system to pump out the exhaust gases to the ambient at higher altitudes is the most complicated process. As Supersonic exhaust diffusers use the momentum of the exhaust gas to reduce the back pressure, they can be considered as the most promising alternative for pumping out the exhaust gases in high altitude test facility. In general, two types of supersonic exhaust diffusers with zero secondary flow injection are used in high test facilities; one being the constant area duct type or the Straight Cylindrical Exhaust Diffuser (SED) and the other one is the variable area duct or the Second Throat supersonic Exhaust Diffuser (STED).

Most of the studies carried out the flow through the convergent divergent nozzle (CD Nozzle) by using a finite volume rewarding code with energy equation, K epsilon viscous model [1]. The increase in the velocity, and decrease in the pressure and temperature are observed all through the CD Nozzle. Convergent divergent nozzle can show better performance than bell nozzle, dual bell nozzle and expandable nozzles [2]. Mach number initially decreases and then increases with the increase in the divergence angle [3] for the supersonic flow in conical nozzle for Mach number 3 at constant throat and inlet diameters. It is also observed that the flow to be super-sonic at exit, decrease in the pressure, increase in the velocity and increase in radial velocity from inlet to outlet is identified

during the analysis of dual bell rocket nozzle. Researches published that the best nozzle performance is obtained for nozzle with 11degrees angle of divergence for both air and gas [4, 5]. So the CD nozzle of 11degrees is the best nozzle to accelerate the flows up to supersonic mach speeds. During simulation analysis on a convergent divergent nozzle k-$\mathcal{E}$ model receives higher average values of Mach number in contrast to k-ω model [6]. Further fluid properties are largely dependent on the cross section of the nozzle which greatly affects the fluid flow [7]. Computational Fluid Dynamic (CFD) simulation results are almost identical to those obtained theoretically [8, 9]. Recent publications report that the continuous increase of nozzle area ratio of satellite launch vehicles for getting the optimum expansion during the mission will contribute up to 20 % of thrust augmentation at low altitudes [10]. When the Flow is from converging nozzle to suddenly expanded circular duct of larger cross-sectional area than that of nozzle exit area, it is found that as the NPR increases, the effect on base pressure is marginal for NPRs up to 2.5; however, at NPR 3 there is a sudden decrease in the base pressure [11] and the viscosity accounts to loss in momentum [12]. It is found that the nozzle-rotor interaction losses can be decreased by selecting appropriate length of axial clearance [13-15]. The CFD model fails to converge if the grid quality is poor and the complexity of the flow near the stagnation region is high [17, 18].

PROBLEM BACKGROUND

An ejector system is designed for the connect pipe facility. The Connect pipe test facility was preliminarily planned with 7.1 kg/s. of Air ejector to simulate 20km altitude condition. During main design phase, with the available theoretical models it was calculated that around 15 kg/s of air is required to simulate 20km altitude. As the Air requirement is more than double and the authenticated data is not available, an experimental, CFD analysis & Thermal analysis plan has been derived to arrive the optimized ejector configuration.

- **Ejector Configuration**

Plenum chamber- diffuser-ejector (PDE) systems are required for high altitude rocket motor test facilities. A 1D Fluid dynamic model was developed to pump out the rocket exhaust from 50 mbar at 20 km altitude to atmospheric condition of 1050mbar. The developed model of a convergent divergent nozzle with SED has the dimensions as mentioned in table 1

Table 1: Ejector Nozzles Details

Name of the Parameter	Value of the Parameter
Inlet diameter of nozzle	76.5mm
Exit diameter of nozzle	76.5mm
Throat diameter of nozzle	23.5mm
Length of the nozzle	325mm
Inlet convergence angle	30^o
Outlet divergence angle	7^o
Length of SED	692mm

- **Nozzle Correlations**

By applying the following correlations to the dimensions of the nozzle the results of various physical parameters are calculated and tabulated in table 2.

$$A_e/A^* = \left(\frac{\gamma+1}{2}\right)^{\frac{-(\gamma+1)}{2(\gamma-1)}} \frac{\left(1+\frac{\gamma-1}{2}M_e{}^2\right)^{\frac{-(\gamma+1)}{2(\gamma-1)}}}{M_e} \tag{1}$$

$$P_e/P_t = \left(1 + \frac{\gamma-1}{2} {M_e}^2\right)^{\frac{-\gamma}{\gamma-1}} \tag{2}$$

$$T_e/T_t = \left(1 + \frac{\gamma-1}{2} {M_e}^2\right)^{-1} \tag{3}$$

$$\dot{m} = \left(\frac{A^* P_t}{\sqrt{T_t}}\right) \sqrt{\frac{\gamma}{R}} \left(\frac{\gamma+1}{2}\right)^{\frac{-(\gamma+1)}{2(\gamma-1)}} \tag{4}$$

$$V_e = M_e \sqrt{\gamma R T_e} \; - \tag{5}$$

Table 2: Results of Nozzle Parameters

Nozzle Parameters	Result Values
$\dot{m}$ (Kg/s)	15
P_e (Pa)	8040Pa
Me	4
To (K)	300
P_o(bar)	7.59
T_e (K)	71.43
Ae/A*	10.719
De (m)	0.184
D* (m)	0.133

METHODOLOGY

The analysis is carried out both with the mathematical and the Computational Fluid Dynamics simulation.

- **Methodology for Mathematical Analysis**

The methodology is mentioned in block diagram as shown in figure 1 has been followed in calculating the nozzle physical parameters using the correlations mentioned from equations (1) to (5).

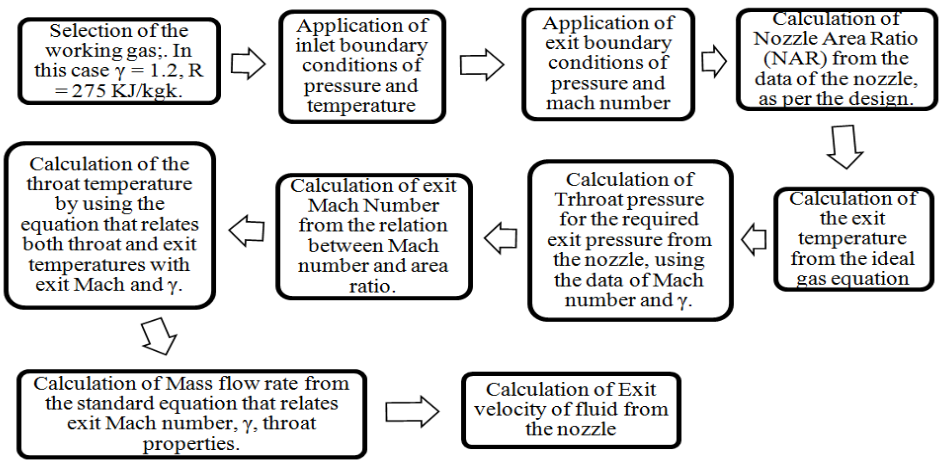


Figure 1: Block Diagram of the Methodology for Mathematical Analysis

- **Computational Fluid Dynamic (CFD) Simulation**

2D Axis-symmetric model has been considered for single stage ejector system CFD simulation. Plenum chamber has been simplified to a small volume region for sake of convergence. A large volume dome atmosphere has been modeled at the exit of mixing chamber to simulate the atmosphere at 1 bar. The geometric model has been prepared in AutoCAD

then meshing, Boundary condition definitions, solver settling and solution has been carried out using ANSYS Cfx. To study the performance of the ejector, the cell pressure and diffuser pressure recovery are to be recorded.

- **Problem Definition**

Flow of gases through a converging-diverging nozzle is one of the benchmark problems used for modeling compressible flow using computational fluid dynamics algorithms. In a converging nozzle, the highest speed that a fluid can be accelerated to is sonic speed, which occurs at the exit. The converging – diverging nozzles are used to accelerate the fluid to supersonic speeds past the throat of such a nozzle. In this case the effect of exit pressure and inlet temperature on the flow through the nozzle and exit Mach number is studied.

The configuration of the nozzle to be studied is a part (primary nozzle) of the two stage ejector designed to carry pay loads (or) exhaust gases. Figure 2 represents the CAD model of the two stage ejector.

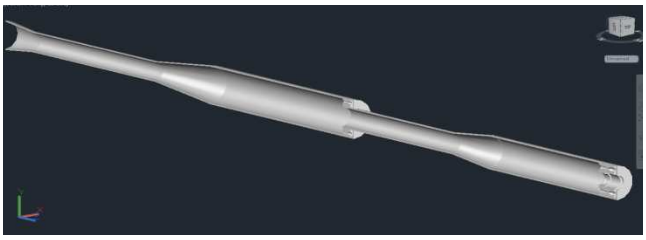
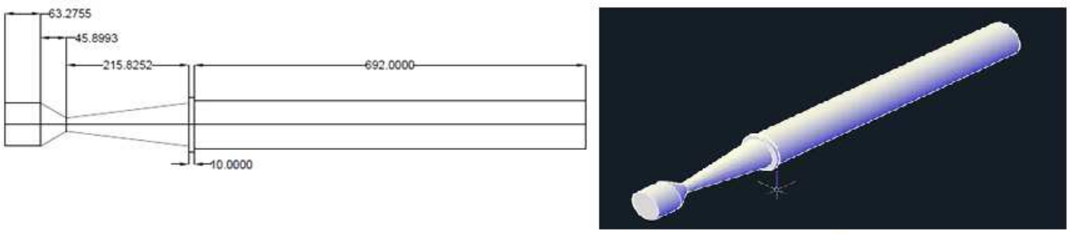


Figure 2: Sectional View of Two Stage Ejector

The geometry of the CD nozzle with SED is fixed (Nozzle Area Ratio-NAR) and the axis-symmetric base model of this study with NAR 10.719 is based on dimensions as shown in figure 3 (a), and the CAD model is developed using Auto CAD and shown in figure 3 (b).



(a) (b)

Figure 3: CD Nozzle with SED Ejector (A) Drafting (B) CAD Model

- **Procedure for CFD analysis in ANSYS**

The procedure as mentioned in figure 4 is followed for the analysis using the ANSYS Software

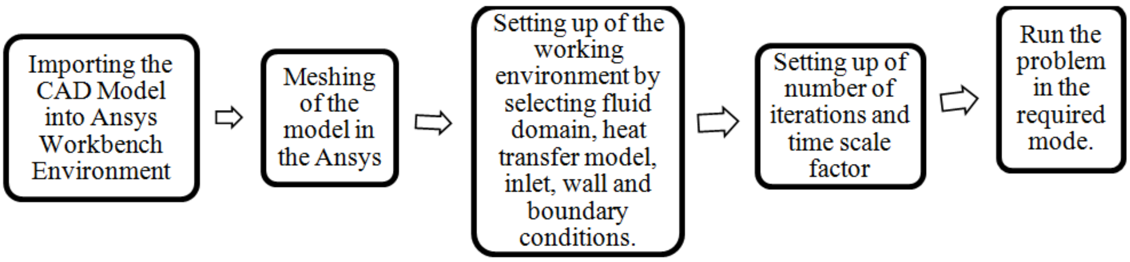


Figure 4: Block Diagram for the Simulation of CFD

RESULTS AND DISCUSSIONS

Firstly the CAD Model is developed in AutoCAD and imported into the ANSYS workbench environment as shown in figure 5 (a). The model is auto meshed with combined element shapes of tetrahedral and quadrilateral as shown in the figure 5(b). After the meshing process, 88356 nodes are formed with 74493 elements. The settings of the environment are considered as the fluid domain with ideal gas air and total energy heat transfer with k-epsilon model. The Boundary walls are to be considered to be adiabatic with no slip condition. Inlet conditions are given to the right end and exit conditions are given to the left end as shown in figure 5(c)

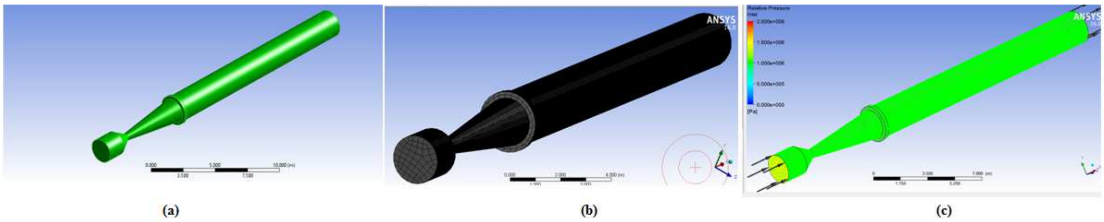


Figure 5: (a) Import of CD Nozzle with SED model (b) Meshing of a CD Nozzle with SED Model (c) Setup for CD Nozzle with SED

The problem has been run with setting up of iterations more than thousand and time scale of 1.0. With the calculated inputs of inlet temperature and exit pressure the program is run in platform MPI local parallel mode with eight partitions. The respective contours are plotted.

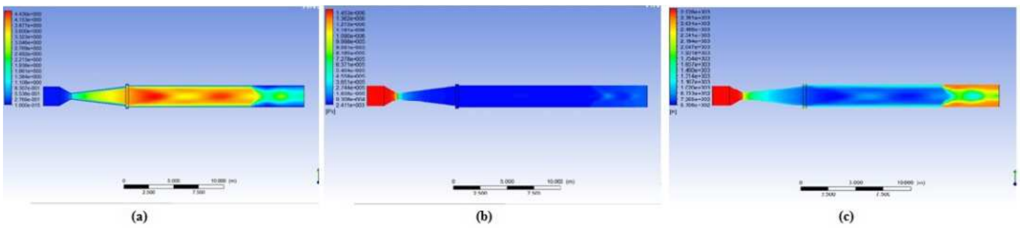


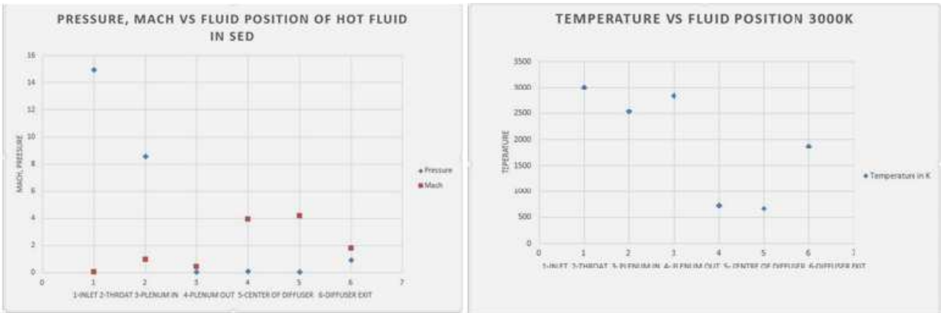
Figure 6: (a) Mach Contour (b) Pressure Contour (c) Temperature Contour of a CD Nozzle with SED Model

Figure 6(a) shows the Mach contours of the model and the values of Mach at inlet, throat, plenum inlet, plenum outlet, center of diffuser, exit of diffuser are 0.055, 0.953, 0.421, 3.916, 4.19 & 1.79 respectively. Figure 6(b) shows the Pressure contours of the CD NOZZLE WITH SED Model and the values of Pressure at inlet, throat, plenum inlet, plenum outlet, center of diffuser, exit of diffuser are 14.96bars, 8.58bars, 0.05423bars, 0.10895bars, 0.06887bars & 0.9357 bars respectively. Figure 6(c) shows the Temperature contours of the SED Model and the values of Temperature at inlet, throat, plenum inlet, plenum outlet, center of diffuser, exit of diffuser are 3000K, 2539.53K, 2836.47K, 737.673K, 662.73K & 1874.71K respectively. The Mach, pressure and temperature variations at different fluid positions along the flow are tabulated in table 3 and are plotted on a graph as shown in Figure 7(a) and 7(b).

Table 3: Mach and Pressure Variation at Different Fluid Positions

	Inlet	Throat	Plenum in	Plenum Out	Centre of Diffuser	Exit
Mach	0.055	0.953	0.421	3.916	4.19	1.796
Pressure(bar)	14.96	8.58	0.0543	0.10895	0.06887	0.9357
Temp (^{O}C)	3000	2539.53	2836.47	737.673	662.73	1874.71

From Figure 7(a) it can be observed that Mach number for a fluid at 3000K on CD nozzle with SED increases from inlet to throat and decreases at plenum in and further increases till center of diffuser and finally decrease at diffuser exit. Also the effect of pressure for a CD nozzle with SED at 3000K is similar to that of Mach number initially the pressure decreases till plenum out later on it compress to the desired diffuser exit pressure. From Figure 7(b) it is implied that unlike pressure and mach number the value of inlet temperature decreases at throat and increases at plenum in gradually the temperature increases till plenum out later on temperature drastically decreases till center of diffuser and attaining required temperature for given output pressure.



(a) **(b)**

Figure 7: (a) Graph for Mach, Pressure (b) Temperature with Respect to Fluid Position

CONCLUSIONS

This project is carried out with boundary conditions of inlet pressure 7.59bar, exit pressure 1050 mbar for CD nozzle with SED for hot fluid of inlet temperature 3000 K. After performing flow analysis with ANSYS Cfx, the following conclusions are made.

- Pressure of fluid decreased and Mach is increased along the flow which facilitates the entrainment of low pressure exhaust gases with the high pressure motive air.
- Further no other fluid is used for the mixing of the fluids to increase the pressure of entrained exhaust gases to the ambient.
- Finally the convergence is achieved with the designed shape of the CD nozzle with SED to drain out the low pressure exhaust gases to the ambient.
- Considerable decrease in the analysis time for convergence is also observed.
- Greater decrease in temperature of the inlet fluid is observed and the rate of decrease of temperature is directly proportional to the inlet fluid temperature. So, ejector system can be used in refrigeration systems namely Ejector Expansion Refrigeration System.

ACKNOWLEDGMENTS

We extend our thanks to principal and management of Institute of Aeronautical Engineering for facilitating the softwares for the successful modeling and Analysis.

NOMENCLATURE

Table 4

NAR	Nozzle Area Ratio	A_e	Exit area of Cross Section
CD	Convergent Divergent	A_*	Throat area of cross section
SED	Straight Cylindrical Exhaust Diffuser	P_t	Total pressure at inlet
STED	Second Throat supersonic Exhaust Diffuser	T_t	Total inlet temperature
NE	Nozzle Exit	$\dot{m}$	Mass flow rate
NPR	Nozzle Pressure Ratio	U_x	Momentum in X direction
ER	Entrainment Ratio	U_y	Momentum in Y direction
CR	Compression Ratio	U_z	Momentum in Z direction
u	Local flow velocity with respect to boundaries	P	Pressure
c	Speed of sound in medium	V	Velocity
V_e	Exit local velocity	h	Enthalpy
M_e	Exit mach number	β	Oblique shock angle
γ	Specific heat ratio	ρ	Density
R	Gas constant for exhaust gases	η	Efficiency
T_e	Exit Temperature		

REFERENCES

1. *Bogdan-Alexandru Belega, Trung Duc Nguyen, 2015, "Analysis Of Flow In Convergent-Divergent Rocket Engine Nozzle Using Computational Fluid Dynamics", International Conference Of Scientific Paper AFASES 2015 Brasov, pp. 28-30.*
2. *Prathibha, M. Satya Narayana Gupta, Simhachalam Naidu, 2015, "CFD Analysis on a Different Advanced Rocket Nozzles", IJEAT, 4(6), pp. 14-22, 2015.*
3. *Pardhasaradhi Natta, V. Rajah Kumar, Dr.Y.V.Hanumantha Rao, 2012 "Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (CFD)", IJERA, 2(5), pp.1226-1235.*
4. *Balaji Krushna.P, P. SrinivasaRao, B. Balakrishna, 2013, "Analysis of Dual Bell Rocket Nozzle Using Computational Fluid Dynamics", IJRET, 2(11), pp. 412-417.*
5. *A. Shanthi Swaroopini, M. Ganesh Kumar, T. Naveen Kumar, 2015, "Numerical Simulation and Optimization Of High Performance Supersonic Nozzle At Different Conical Angles", IJRET, 4(9), pp. 268-273.*
6. *Omid Joneydi Shariatzadeh, Afshin Abrishamkar, and Aliakbar Joneidi Jafari, 2014, "Computational Modelling of a Typical Supersonic Converging-Diverging Nozzle and Validation by Real Measured Data".*
7. *G. Satyanarayana, Ch. Varun, S.S. Naidu, 2013, " CFD Analysis of Convergent-Divergent Nozzle", Acta Technica Corviniensis-Bulletin Of Engineering Tome VI – Fascicule 3.*
8. *CH. Srinivasa Chakravarthy1, R.Jyothu Naik, 2015, "Analysis Of Flow In A De Laval Nozzle Using Computational Fluid Dynamics", Proceedings of International Conference on Recent Trends in Mechanical Engineering-2K15(NECICRTME-2K15), pp. 2454-9614.*
9. *M. Sundararaj and S. Elangovan, 2013, "Computational Analysis of Mixing Characteristics of Jets from Rectangular Nozzle with Internal Grooves", Indian Journal of Science and Technology, 6(5S), pp. 4543-4548.*
10. *Saravanan Manikrishnan. P. Adhavan, C. Boobala Karthikeyan, Mohanraj Murugesan, and Sanal Kumar.V.R., 2014, "Numerical Studies on Altitude Compensation Nozzles for Aerospace Vehicles", 3rd International Conference on Mechanical, Automobile and Robotics Engineering (ICMAR'2014)*

11. *Syed Ashfaq, 2014, "Studies On Flow From Converging Nozzle And The Effect Of Nozzle Pressure Ratio For Area Ratio Of 6.25", IJESAT, 4(1), pp. 49-60.*

12. *Nadeem N, Dandotiya D, Najar F, 2013, "Modeling & Simulation of Flow Separation & Shocks in a CD Nozzle", IJMERA, 1(3), pp. 14-21.*

13. *Ms. B.Krishna Prafulla1, Dr. V. Chitti Babu and Sri P. Govinda Rao, 2013, "CFD Analysis of Convergent- Divergent Supersonic Nozzle", IJCER, 3(5), pp. 5-16.*

14. *Jean-Baptiste Mulumba Mbuyamba, 2013, "Calculation And Design Of Supersonic Nozzles For Cold Gas Dynamic Spraying Using Matlab And Ansys Fluent", Thesis, University of the Witwatersrand, Johannesburg.*

15. *Karla Keldani Quintão, 2012, "Design Optimization Of Nozzle Shapes For Maximum Uniformity Of Exit Flow", FIU Electronic Theses and Dissertations, Paper 779.*

16. *Ekanayake, E. M. Sudharshani, 2013, "Numerical Simulation of a Convergent Divergent Supersonic Nozzle Flow", Dissertation, RMIT University, Melbourne, Australia.*

17. *Srikrishna C. Srinivasa, 2012, "CFD Modeling and Analysis of an Arc-jet facility using ANSYS Fluent", Thesis, San José State University, San Jose, CA-95192.*

18. *Marc Linares, Alessandro Ciampitti Marco Robaina, 2015, "Design Optimization of Supersonic Nozzle", FIU.*

AUTHOR PROFILE

Dr. CH V K N S N Moorthy awarded his Ph.D in Mechanical Engineering from GITAM University, Vishakhapatnam, Andhra Pradesh, India. Presently he is working as Professor of Mechanical Engineering at Institute of Aeronautical Engineering (Autonomous), Dundigal, Hyderabad, Telangana State, India. His research interests are in heat transfer, nano technology and CFD.

Dr. V. Srinivas is presently working as a Professor of Mechanical Engineering at GITAM University Vishakhapatnam, Andhra Pradesh, India. His research interests are nanofluids for lubricant & coolant applications and nano coatings for IC engines.

Prof. V V S H Prasad is presently working as a professor of Mechanical Engineering at Institute of Aeronautical Engineering (Autonomous), Dundigal, Hyderabad, Telangana State, India. He worked as a General Manager at Praga Tools Limited, Hyderabad,India in the research and development division recognized by DST, Govt. of India for import substitution. His research interests are CAD/CAM and machine design.

Ms. T. Vanaja is presently working as an Assistant Professor of Mechanical Engineering at Institute of Aeronautical Engineering (Autonomous), Dundigal, Hyderabad, Telangana State, India. Her research interests are CAD/CAM and WEDM.